

A Zonal Turbulence Modeling Approach for ICE Flow Simulation

Vesselin Krassimirov Krastev
University of Tuscia

Gino Bella
University of Rome "Tor Vergata"

ABSTRACT

Turbulence modeling is a key aspect for the accurate simulation of ICE related fluid flow phenomena. RANS-based turbulence closures are still the preferred modeling framework among industrial users, mainly because they are robust, not much demanding in terms of computational resources and capable to extract ensemble-averaged information on a complete engine cycle without the need for multiple cycles simulation. On the other hand, LES-like approaches are gaining popularity in recent years due to their inherent scale-resolving nature, which allows the detailed modeling of unsteady flow features such as cycle-to-cycle variations in a DI engine. An LES requires however a large number of simulated engine cycles to extract reliable flow statistics, which coupled to the higher spatial and temporal resolution compared to RANS still poses some limits to a wider application of such methodology on realistic engine geometries. In this paper a hybrid zonal RANS/LES simulation methodology is proposed, based on a Detached Eddy Simulation (DES) technique previously developed by the authors. The aim is to preserve the turbulence scale-resolving capabilities wherever actually needed, reducing at the same time the overall computational costs compared to standard LES. The resulting zonal model has been implemented into the open-source CFD package OpenFOAM® and then assessed on geometries which reproduce the typical flow conditions at cylinders' intake ports. The presented numerical solutions have been compared with the available experimental benchmarks as well as with previous computational studies from other authors.

CITATION: Krastev, V. and Bella, G., "A Zonal Turbulence Modeling Approach for ICE Flow Simulation," *SAE Int. J. Engines* 9(3):2016, doi:10.4271/2016-01-0584.

INTRODUCTION

The current standard for turbulence modeling in the CFD simulation of Internal Combustion Engines (ICE) is still represented by RANS-based closures [1, 2, 3, 4]. Advantages of Unsteady-RANS (URANS) modeling are well known and include an acceptable level of accuracy obtained at relatively low computational costs, as well as the capability of readily providing turbulence-related scalar quantities for other physical sub-models (e. g. combustion). On the other side, due to the purely modeled nature of the turbulent fluctuations spectrum, URANS methods are also inherently not able to predict small flow scales as well as large scale cyclic variations inside the piston-cylinder assembly.

To overcome the shortcomings of URANS, Large Eddy Simulation (LES) is the natural step ahead. With LES most of the turbulent flow scales are actually resolved, while only the effect of smaller structures (i. e. the ones below an implicitly or explicitly specified filtering threshold) is modeled. This offers significant theoretical advantages for ICE applications, such as the accurate description of flow and combustion cycle-to-cycle variability [5,6]. Early incylinder LES studies date back to 1992 [7,8], but since then the number of reported applications of practical interest has increased quite slowly (see Figure 1). One of the commonly accepted explanations for this

slow progress is that the computational requirements of a standard well resolved LES still restrict its usage as an engine design and optimization tool [6]. In fact, although the typical grid size adopted in URANS in-cylinder simulations is usually fine enough to resolve about 80% of the flow's kinetic energy [7], time steps adopted in LES are much smaller compared to URANS. Moreover, the need to resolve a statistically relevant number of cycles (at least of the order of 10) can push the cpu time of an LES up to 2 orders of magnitude beyond what is needed for a URANS. Other concerns, more generally related to the application of LES to complex geometries, include near-wall resolution and the consistent definition of actual turbulent fluctuations at the domain inlets [31].

The issues mentioned above can be possibly alleviated by hybrid URANS/LES methods. Although conceptually not new [9] and already quite popular for the simulation of other types of internal combustion devices [10], this class of methods is still relatively unexplored in the reciprocating engines modeling field. As shown in Figure 1, only a few relevant URANS/LES ICE applications can be reported in the scientific literature up to 2015 [11, 12, 13, 14, 15]. Nonetheless, all of them have already shown promising capabilities in the description of large-scale flow oscillations for simplified tumbling motion setups, fixed intake ports and even complete motored piston-cylinder assemblies. For instance, in the context of

al. [15] a filtering procedure is directly applied to the RANS modeled turbulent quantities, obtaining as-good-as-LES description of large scale flow structures but on coarser grids and with much simpler boundary conditions. In several other works [11,12,13], the Detached Eddy Simulation (DES) technique has been applied. The DES original concept relies on the simulation of attached flow regions through URANS, thus bypassing near-wall LES resolution requirements, with a seamless URANS-to-LES transition once abrupt separations occur [16]. This has been found to be fairly efficient in ICE-like geometries [11,12], but has also evidenced some limitations due to potential inaccuracies in the URANS-to-LES automatic switching in complex internal flow configurations. These inaccuracies usually manifest themselves as the so called Modeled Stress Depletion (MSD) or as a slow shear-layer development [13,16].

Seamless DES is non-zonal, meaning that the turbulence treatment choice is not managed by CFD users but by the model itself. Conversely, if a Zonal-DES (ZDES) method is applied different subdomains are explicitly marked as URANS, LES or DES by the user, thus fully clarifying the expected behavior of the flow [16]. This approach can be very effective, but it poses the question of how to optimize the regions' distribution in non-trivial geometries and flow configurations. In principle, typical ICE computational domains allow a relatively easy zonal breakdown among different regions (e. g. in-cylinder volume, intake and exhaust ducts, plenums) representing therefore a potentially good application for ZDES.

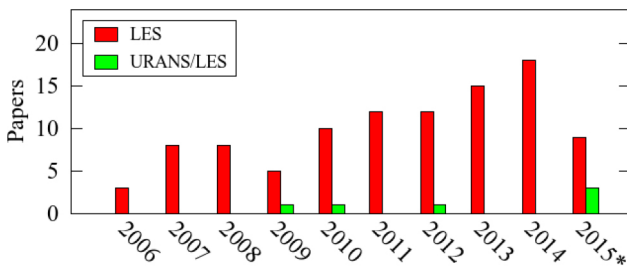


Figure 1. Per-year number of published papers with relevant LES and hybrid URANS/LES ICE flow applications (source: www.scopus.com; *year 2015 data are provisional)

In order to evaluate if ZDES is suitable for ICE multidimensional simulations, in the present contribution we assess a fully zonal variant of a previously proposed DES model [13]. The paper is organized as follows: first, the model derivation is briefly described including details on its calibration in pure LES mode; subsequently, two standard ICE-like flow benchmarks are addressed, comparing the numerical predictions with the reference experimental data as well as with prior computational studies performed on the same configurations; finally, at the end of the results sections, concluding remarks and proposals for future developments are added.

THE K-G ZDES MODEL

Derivation

Our original non-zonal DES proposal is based on an improved version of the so called k-g turbulence model [17,18]. In its RANS form, it employs the standard eddy-viscosity closure assumption and consists of the following transport equations (valid for a constant-density flow):

$$\frac{\partial k}{\partial t} + \nabla \cdot (\mathbf{U}k) = \mathcal{P} - \frac{k}{\tau} + \nabla \cdot \left[\left(\frac{\nu_t}{\sigma_k} + \nu \right) \nabla k \right] \quad (1)$$

$$\frac{\partial g}{\partial t} + \nabla \cdot (\mathbf{U}g) = \frac{\beta g}{2\beta^* \tau} - \frac{\alpha g^3}{2k\tau} \mathcal{P} + \nabla \cdot \left[\left(\frac{\nu_t}{\sigma_g} + \nu \right) \nabla g \right] - \left(\frac{\nu_t}{\sigma_g} + \nu \right) \frac{3g}{\tau} (\nabla g \cdot \nabla g) \quad (2)$$

$$\nu_t = \beta^* k \tau \quad (3)$$

where, in three dimensions [18]:

$$\tau = \min(g^2, \alpha \tau_{lim,3D}); \quad \tau_{lim,3D} = \frac{2}{3\beta^*} \sqrt{\frac{3}{8|\mathbf{E}|^2}} \quad (4)$$

with $|\mathbf{E}|^2$ = magnitude squared of the mean rate-of-strain tensor and $\alpha < 1$ being the bound tuning constant (usually set around 0.6 [18]). To obtain a two-equation DES form, attention should be focused on the turbulent kinetic energy destruction term [19], which in Equation (4) is given by:

$$\mathcal{S}_{k,RANS} = \frac{k}{\tau} = \frac{k^{3/2}}{l_{k-g}}; \quad l_{k-g} = k^{1/2} \tau \quad (5)$$

A straightforward $\mathcal{S}_{k,DES}$ expression will be then given by:

$$\mathcal{S}_{k,DES} = F_{DES} \cdot \mathcal{S}_{k,RANS}; \quad F_{DES} = \max \left(\frac{l_{k-g}}{C_{DES} \Delta}, 1 \right) \quad (6)$$

with Δ = local grid spacing parameter and $C_{DES} \approx \mathcal{O}(1)$. To prevent an inconsistent near-wall flow resolution and the so-called grid-induced separation, a delaying function γ_d should be added as follows [20]:

$$\mathcal{S}_{k,DDES} = F_{DDES} \cdot \mathcal{S}_{k,RANS}; \quad F_{DDES} = \max \left[\gamma_d \left(\frac{l_{k-g}}{C_{DES} \Delta} \right), 1 \right] \quad (7)$$

$$\gamma_d = 1 - \tanh \left[\left(8 |r_d| \right)^3 \right]; \quad r_d = \frac{\nu_t + \nu}{\sqrt{2\mathbf{E} : \mathbf{E}} \kappa^2 y^2} \quad (8)$$

With $\kappa = 0.41$ (V. Karmán constant) and y = wall-normal distance.

Inserting expression (11) into Equation (1) yields the baseline non-zonal k-g Delayed-DES (DDES) turbulence model [23]. At this point, zonalization can be easily achieved by introducing two boolean

switching parameters, C_{z1} and C_{z2} . The first one determines if the standard DDES or the zonal URANS/LES formulation will be used, by substituting F_{DDES} with:

$$F_{DDES}^* = C_{z1} \cdot F_{DDES} + (1 - C_{z1}) F_{ZDES} \quad (9)$$

where F_{ZDES} is defined by the second parameter as follows:

$$F_{ZDES} = C_{z2} + (1 - C_{z2}) \cdot \left(\frac{l_{k-g}}{C_{DES} \cdot \Delta} \right) \quad (10)$$

Table 1 resumes all the possible behaviors produced by F_{DDES}^* , depending on whether C_{z1} and C_{z2} are equal to 0 or 1.

Table 1. k-g model zonalization determined by the different C_{z1} and C_{z2} values.

$C_{z2} \backslash C_{z1}$	1	0
1	DDES	RANS
0	DDES	LES

LES Mode Calibration

Convection Scheme Choice

It has been already argued by previous researchers [19] that a key factor for a successful hybrid approach is represented by a proper numerical treatment of the nonlinear convective transport terms in the discretized momentum equations. More specifically, low-dissipation centered schemes should be employed in the LES zones of the flow, while for the RANS regions upwind-biased schemes are better suited due to the typical stability issues of high local Reynolds number flows [21]. In the following we show the procedure adopted for the optimal LES convection scheme choice.

The model has been numerically implemented within the open-source CFD toolbox OpenFOAM®. The OpenFOAM® code is designed to operate in a generic finite volume discretization background, with arbitrarily shaped polyhedral cells and a collocated arrangement of the flow variables. The Reynolds-Averaged or filtered Navier Stokes equations are solved through a generalized pressure-velocity coupling procedure. In steady-state mode that acts like the SIMPLE algorithm [21], while in unsteady mode it turns into Issa's non-iterative PISO method [22].

In our numerical tests, we have assumed the kinetic energy conservation as the main scheme evaluation parameter. Energy conservation is a feature typically encountered in high-order centered schemes and essentially means that in a zero-viscosity flow the spatially averaged kinetic energy should not be numerically dissipated over time. We defined an idealized "turbulence box" test case where the grid consists of 64^3 perfectly cubic cells, with cyclic

boundary conditions in all directions. The bulk velocity field is initialized through a divergence-free three dimensional turbulent energy spectrum, following the generalized spectral function proposed by [23]. Navier-Stokes equations are discretized and solved through time without any explicit filtering or averaging and, in order to evidence the dissipation introduced by numerics, a perfectly incompressible zero-viscosity flow is considered. Time integration is performed by implementation of a second-order accurate three-point implicit backward scheme.

Among the various alternatives available in OpenFOAM®, the following momentum convection schemes have been considered:

1. standard second-order Central Differencing (CD);
2. Linear Upwind-Stabilized Transport (LUST);
3. high-wavenumber Filtered Central Differencing (FCD).

The LUST scheme is a 75%-25% blend between CD and second-order Linear Upwind (LU), developed to introduce upwind biased stabilization of grid-scale oscillations [24]. FCD is designed to remove high-frequency modes in the numerical solution by comparing the face gradient with neighboring cell gradients and then limiting their ratio [24]. The OpenFOAM® FCD implementation has a scaling parameter for the gradient ratio which ranges from 0 to 1, with 0 corresponding to standard CD. For testing purposes, we considered here values ranging from 0.25 to 1.

In Figures 3 and 4 we show the results of our investigation in terms of the non-dimensional time coordinate t/t_{k0} , with t_{k0} given by:

$$t_{k0} = t \cdot \lambda_{max} \cdot (k_{avg,0})^{(1/2)} \quad (11)$$

Due to the upwind biasing the LUST scheme exhibits a clearly overdissipative behavior, while as expected CD is nearly energy-conserving. FCD stands in between showing a dependency from the scaling parameter value. Consequently, we have chosen the FCD scheme with a parameter of 0.25, thus accepting a small increase in energy dissipation in favor of an improved stability compared to standard CD.

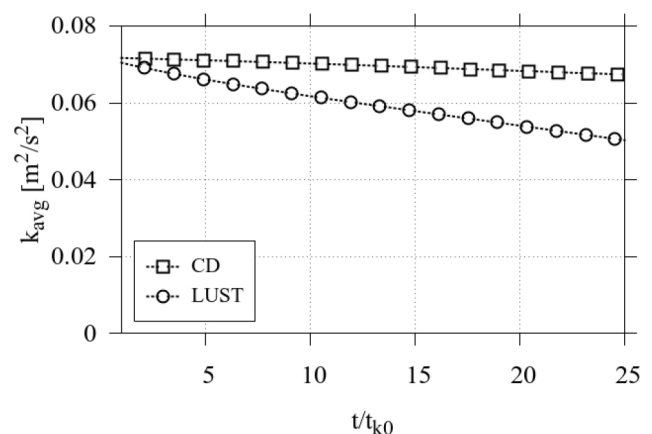


Figure 2. Kinetic energy temporal evolution for the turbulence box test case vs. LUST.

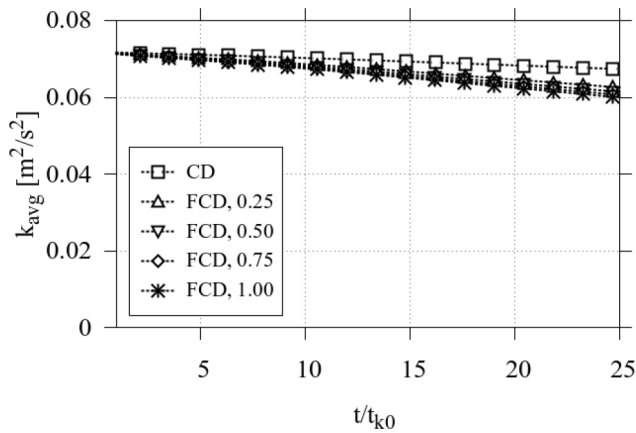


Figure 3. Kinetic energy temporal evolution for the turbulence box case: CD vs. FCD.

C_{DES} Constant Calibration

The C_{DES} model constant is directly linked to the grid-dependent length scale Δ and the combination of the two should be able to provide a consistent turbulent energy decay in LES mode. A common assumption for Δ , which has also been adopted by previous DES researchers (see e. g. [25]), is the following:

$$\Delta = (\Delta V)^{(1/3)} \quad (12)$$

where ΔV is the volume of each computational cell.

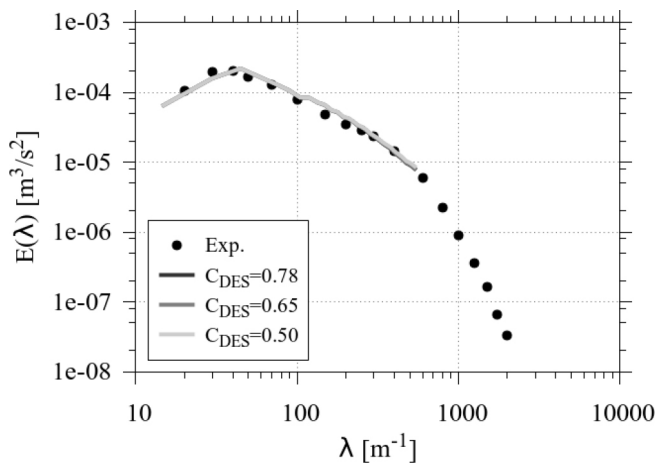


Figure 4. Three-dimensional energy spectra at $t = t_1$.

The energy decay tests have been carried out on the previously described turbulence box case, comparing the predicted turbulent spectra with the experimental database of Comte-Bellot and Corrsin [26]. The highest resolvable wavenumber was set at approximately $5.3 \cdot 10^2 \text{ m}^{-1}$ and the velocity field was initialized fitting the reference spectral function to the conditions detected at the first measuring station in the experiments. The flow was then left to a spontaneous decay up to the conditions corresponding to the second and third stations ($t = t_1$ and $t = t_2$). Fluid kinematic viscosity was set considering air at ambient condition, while the k-g ZDES formulation was run in pure LES sub-grid scale mode ($C_{z1} = C_{z2} = 0$). Initial bulk

values for k and g were imposed in order to have a viscosity ratio of 1/10, whereas their respective transport equations were discretized using a bounded second-order LU scheme for convection. For the momentum convection we have used FCD 0.25 and all the other numerical settings are the same as adopted for the energy conservation tests.

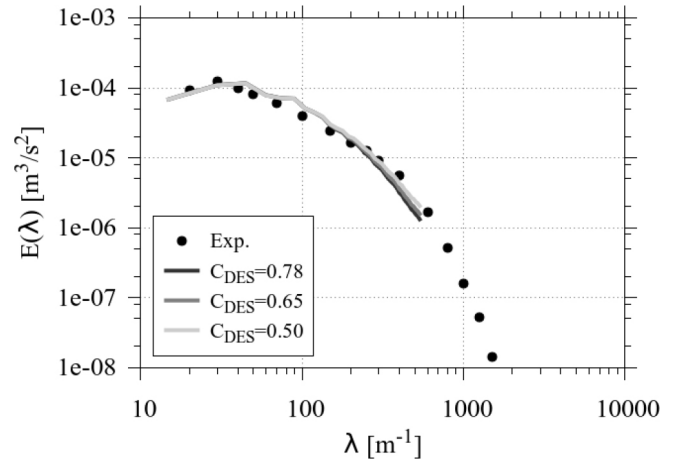


Figure 5. Three dimensional energy spectra at $t = t_2$.

C_{DES} was initially set to 0.78 [19] and after that it was lowered to 0.65 and 0.5. Figures 4 and 5 collect the predicted three-dimensional spectra at the two reference simulation times for all the considered C_{DES} values. At $t = t_1$, there is little difference between the reported trends. At $t = t_2$, $C_{DES} = 0.5$ emerges as the value that best matches the experimental measurements. The latter was then assumed as default for all the simulations shown in the next sections.

SUDDEN EXPANSION FLOW

For the initial testing of the k-g ZDES model, a sudden circular expansion flow was considered (Figure 6). In spite of its relative simplicity, this case reproduces well one of the main ICE flow complexities which is the abrupt geometry-induced separation at the cylinder's intake ports. The modeled configurations follows the experimental setup of Dellenback et al. [27], which includes the effects of an imposed swirling motion at the inlet.

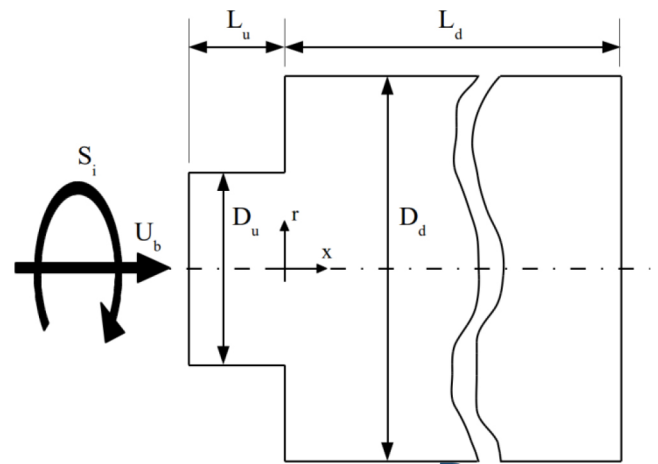


Figure 6. Sketch of the sudden expansion flow (dimensions are not in scale).